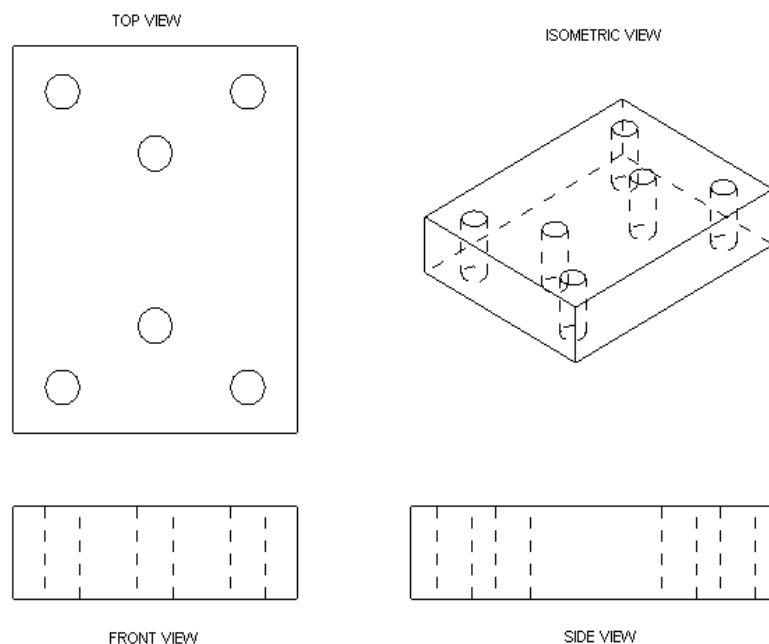


## CHAPTER 5 – DRAFTING

The NX 12 *Drafting* application lets you create drawings, views, geometries, dimensions, and drafting annotations necessary for manufacturing as well as understanding of an industrial design. The goal of this chapter is to give the designer/draftsman enough knowledge of drafting tools to create a basic drawing of their design. The drafting application supports the drafting of engineering models in accordance with ANSI standards. After explaining the basics of the drafting application, we will go through a step-by-step approach for drafting some of the models created earlier.

### 5.1 OVERVIEW

The *Drafting* Application is designed to allow you produce and maintain industry standard engineering drawings directly from the 3D model or assembly part. Drawings created in the *Drafting* application are fully associative to the model and any changes made to the model are automatically reflected in the drawing. The *Drafting* application also offers a set of 2D drawing tools for 2D centric design and layout requirements. You can produce standalone 2D drawings. The *Drafting* Application is based on creating views from a solid model as illustrated below. *Drafting* makes it easy to create drawings with orthographic views, section views, imported view, auxiliary views, dimensions and other annotations.



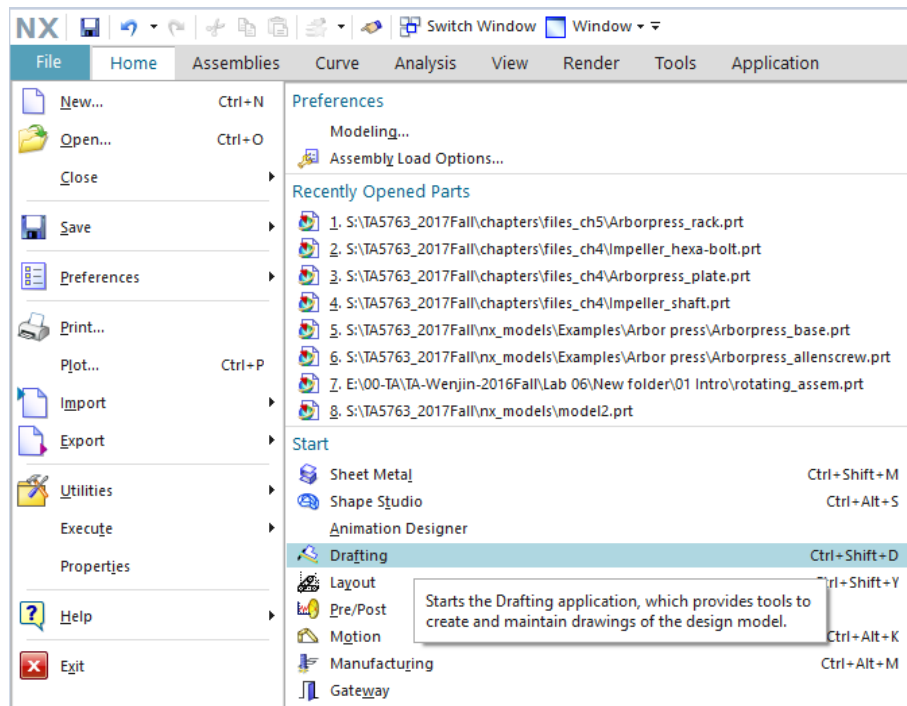
Some of the useful features of the *Drafting Application* are:

- 1) After you choose the first view, the other orthographic views can be added and aligned with the click of a few buttons.
- 2) Each view is associated directly with the solid. Thus, when the solid is changed, the drawing will be updated directly along with the views and dimensions.
- 3) Drafting annotations (dimensions, labels, and symbols with leaders) are placed directly on the drawing and updated automatically when the solid is changed.

We will see how views are created and annotations are used and modified in the step-by-step examples.

## 5.2 CREATING A DRAFTING

- Open the file **Arborpress\_rack.prt**
- From the NX 12 Interface, choose **File** → **Drafting** as shown or choose **Application** tab and select **Drafting**



When you first open the *Drafting Application*, a window pops up asking for inputs like the *Template*, *Standard Size* or *Custom Size*, the units, and the angle of projection.

### Size

*Size* allows you to choose the size of the *Sheet*. There are standard *Templates* that you can create for frequent use depending upon the company standards. There are several *Standard* sized *Sheets* available for you. You can also define a *Custom* sized sheet in case your drawings do not fit into a standard sized sheet.

### Preview

This shows the overall design of the *Template*.

### Units

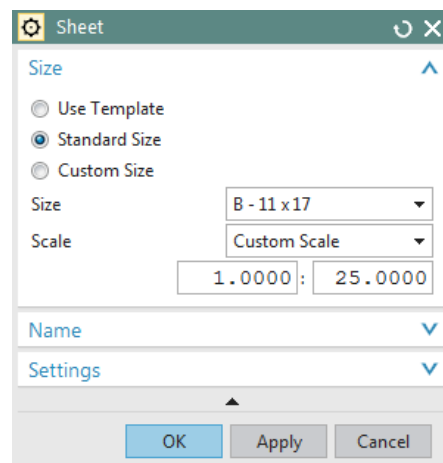
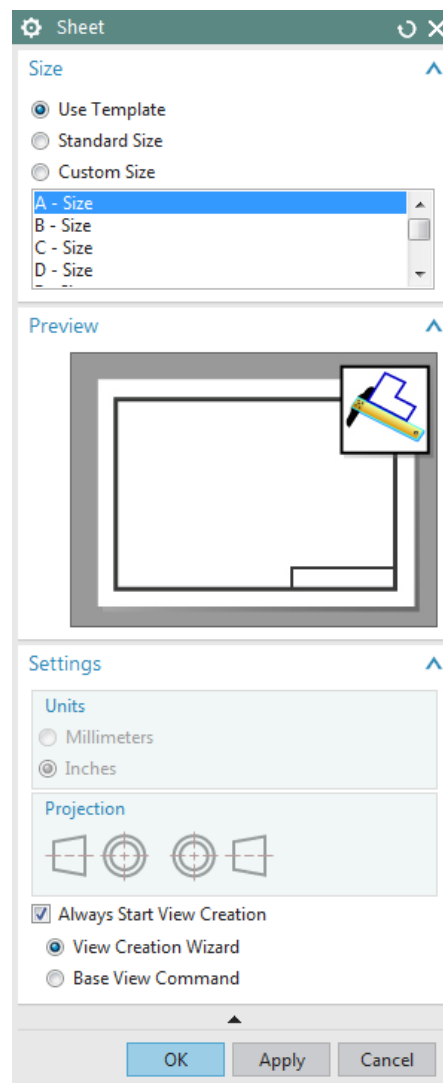
*Units* follow the default units of the parent 3-D model. In case you are starting from the *Drafting Application* you need to choose the units here.

### Projection

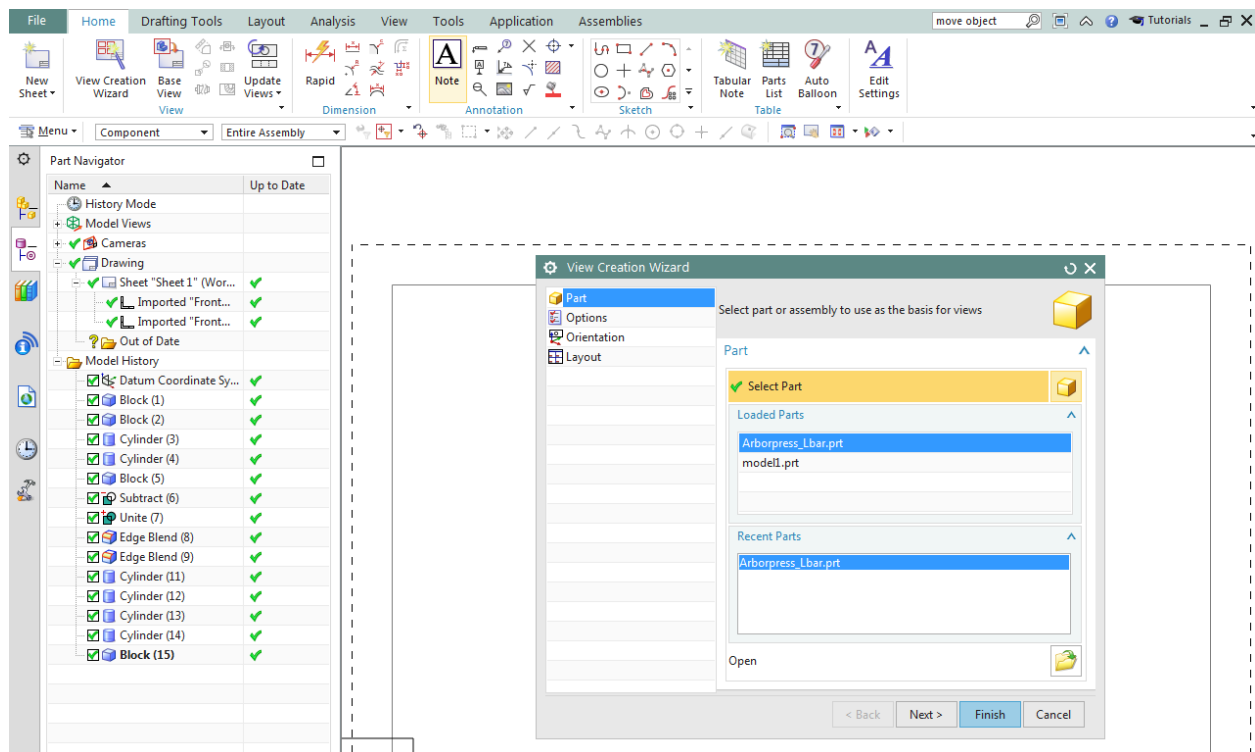
You can choose the *Projection Method* either *First Angle* or *Third Angle* method.

To start using the *Drafting Application* we will begin by creating a *Standard Sized* sheet:

- Click on the **Standard Size** radio button
- In the drop-down menu on the **Size** window, select **sheet B**, which has dimensions **11 x 17**
- Change the **Scale** to **1:25** by using the drop-down menu and choosing the **Custom Scale** under the **Scale**
- Click **OK**

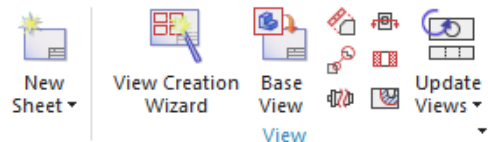


This will open the *Drafting Application* and the following screen will be seen as below. Let us first look at the *Drafting Application Interface*.



You will see a dialog box pops-up which will help you choose the parts, views and other options.

- Change the options and views and click **Finish**
- Choose **Insert** → **View** → **Base** or click on **Base View** in the **View Group**




The *Base View* dialog box with the options of the **View** and the **Scale** will show up along with a floating drawing of the object.

- Choose the **View** to be **Front**

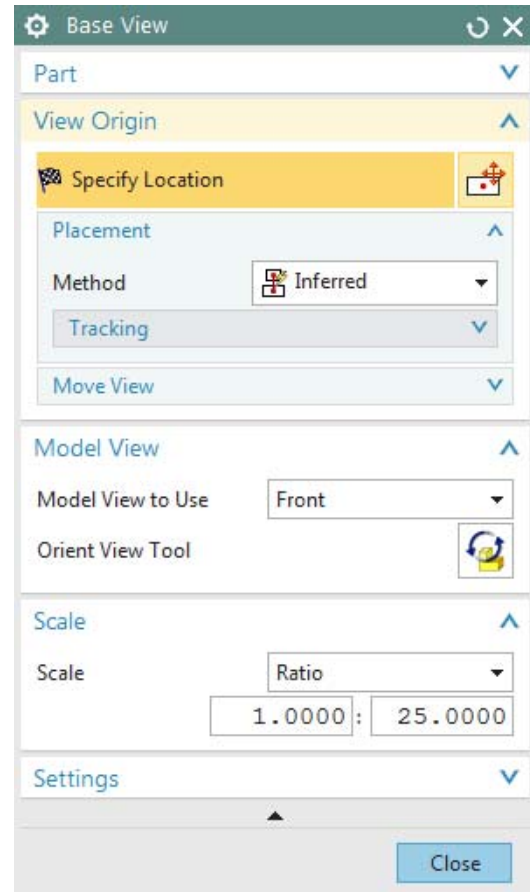
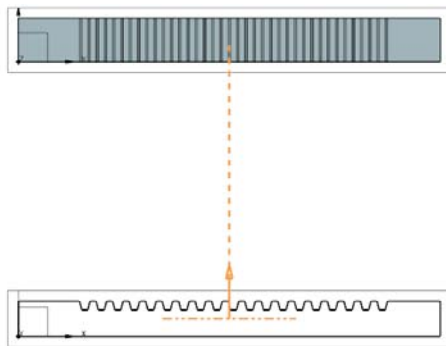
You can find the *Front View* projection on the screen. You can move the mouse cursor on the screen and click on the place where you want the view.

Once you set the *Front View* another dialog box will pop-up asking you to set the other views at any location on the screen within the *Sheet Boundary*.

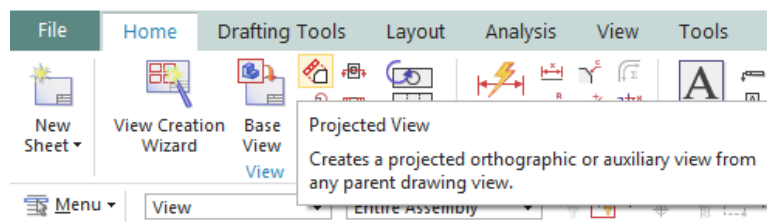
You can find different views by moving the cursor around the first view. If you want to add any orthographic views after closing this file or changing to other command modes

- Choose **Insert** → **View** → **Projected View** or choose **Projected View**  icon from the **View** group

Now, let us create all the other orthographic projected views and click on the screen at the desired position.




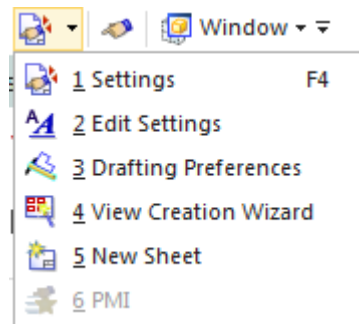
- In case you have closed the **Projected View** dialog box you can reopen it by clicking on the **Projected View** icon in the **View Group**



- Move the cursor and click to get the other views
- Click **Close** on the **Projected View** dialog box or press **<Esc>** key on the keyboard to close the window

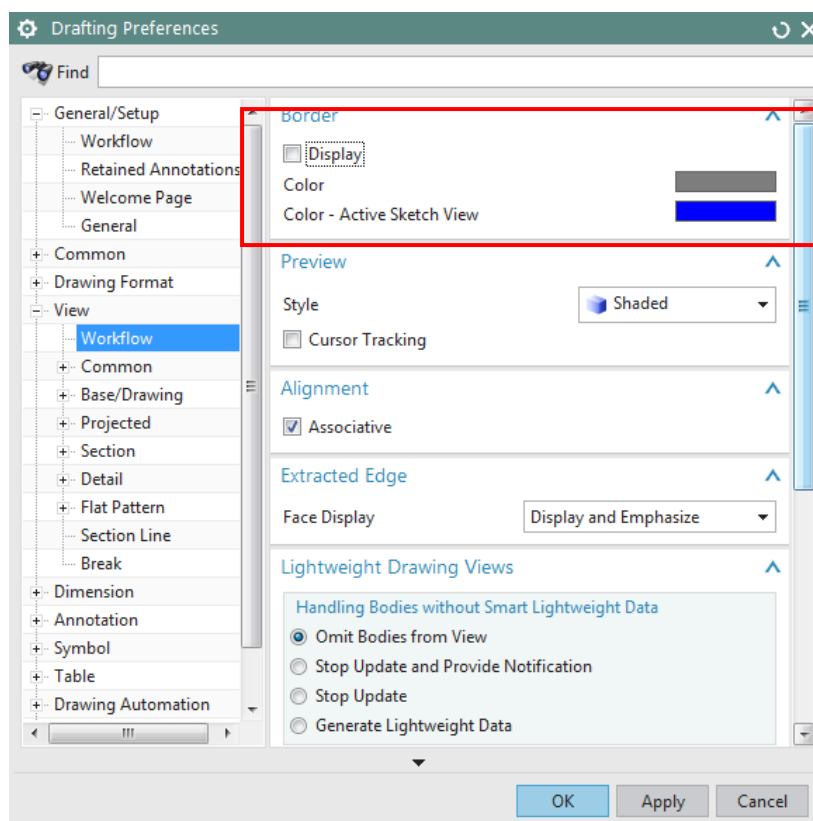
Before creating the dimensions, let us remove the borders in each view as it adds to the confusion with the entity lines.

- Choose **Menu** → **Preferences** → **Drafting** or click on  icon in the **Quick Access** toolbar to find the **Drafting Preferences**



The *Drafting Preferences* window will pop up.

- Click on the **VIEW** tab button
- Uncheck the **Tick** mark on the **Display Borders** as shown in the figure below and click **OK**



There are many other options like number of decimal places, hidden lines, angles, and threads that you can find here. For example, you can find options for hidden lines in *Drafting Preferences* → *View* → *Common* → *Hidden Lines*

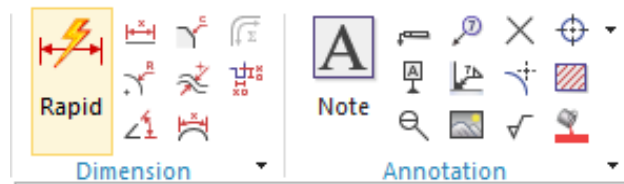
## 5.3 DIMENSIONING

Now let's move on to create the dimensions for these views. The dimensions can be inserted by either of the two ways as described below:

- Choose **Menu** → **Insert** → **Dimension**

OR

- Click on the **Dimension Toolbar** as shown in the following figure

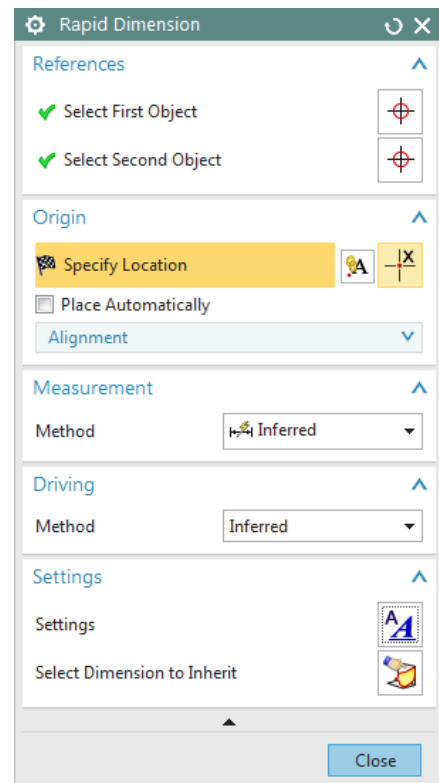
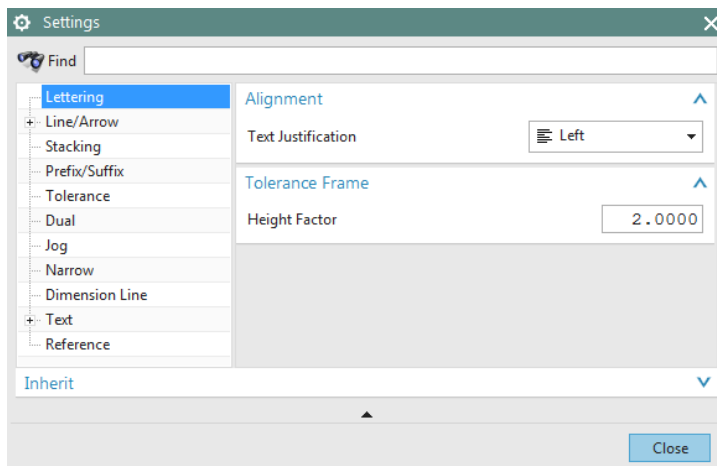


- Click on **Points** and **Edges**, move the mouse and click on the appropriate location to draw dimensions

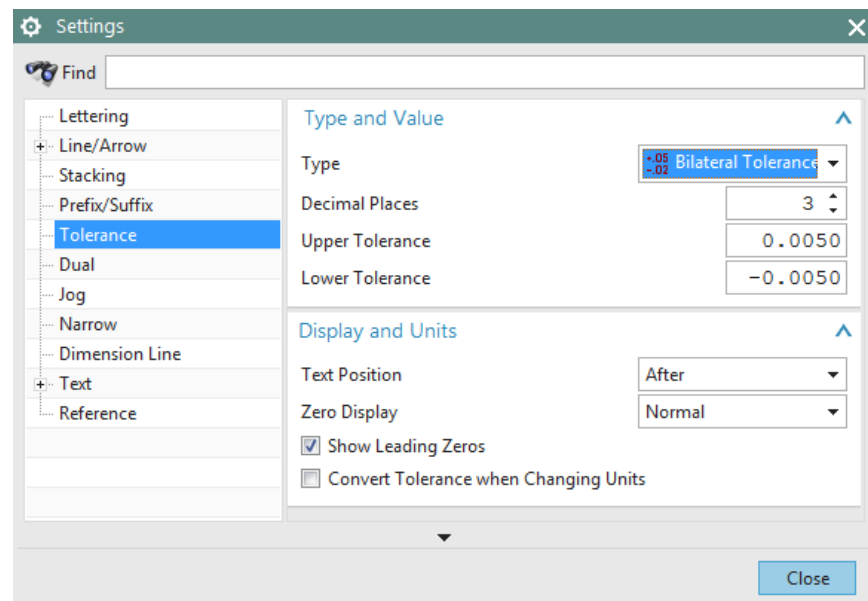
The icons in this window are helpful for changing the properties of the dimensions.

- Click on the **Settings Button**

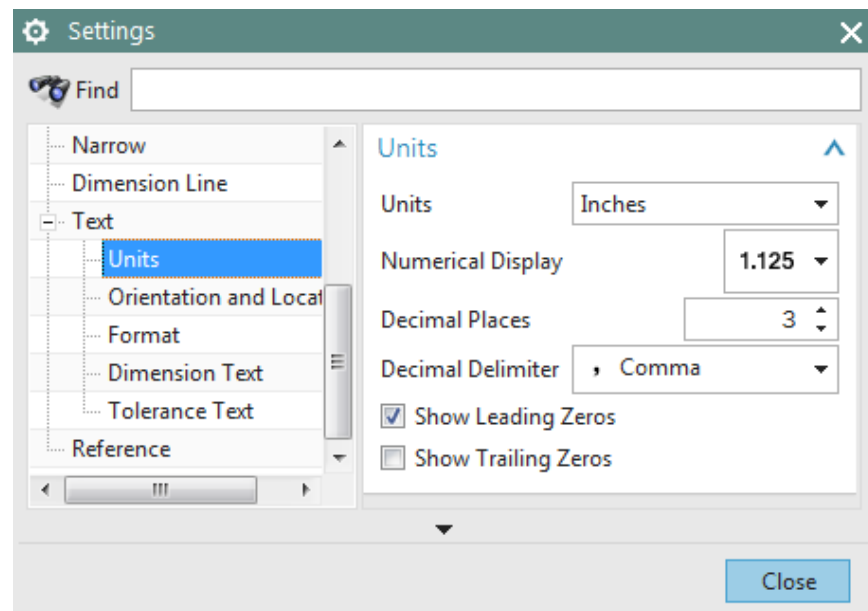
Here you will be able to modify the settings for dimensioning. A dialog appears as shown below.



The first list is for *Lettering*. This allows the user to justify and select the frame size. In the *Line/Arrow* section, you can vary the thickness of the arrow line, arrow head, angle format etc. The most important section is the *Tolerance* list. Here you can vary the tolerance to the designed value.

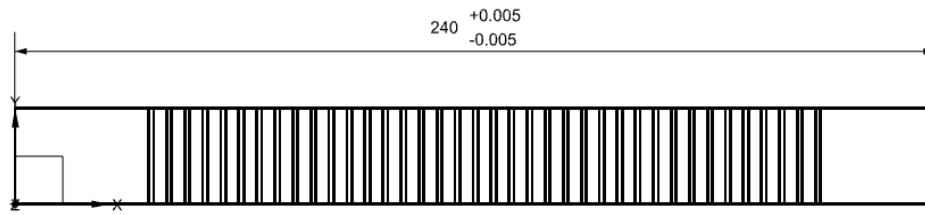


The type of display, precision required for the digits and other similar options can be modified here. The next icon is the *Text* option, which you can use to edit the units, text style, font and other text related aspects.





- On the first view (**Front View**) that you created, click on the top left corner of the rack and then on the top right corner

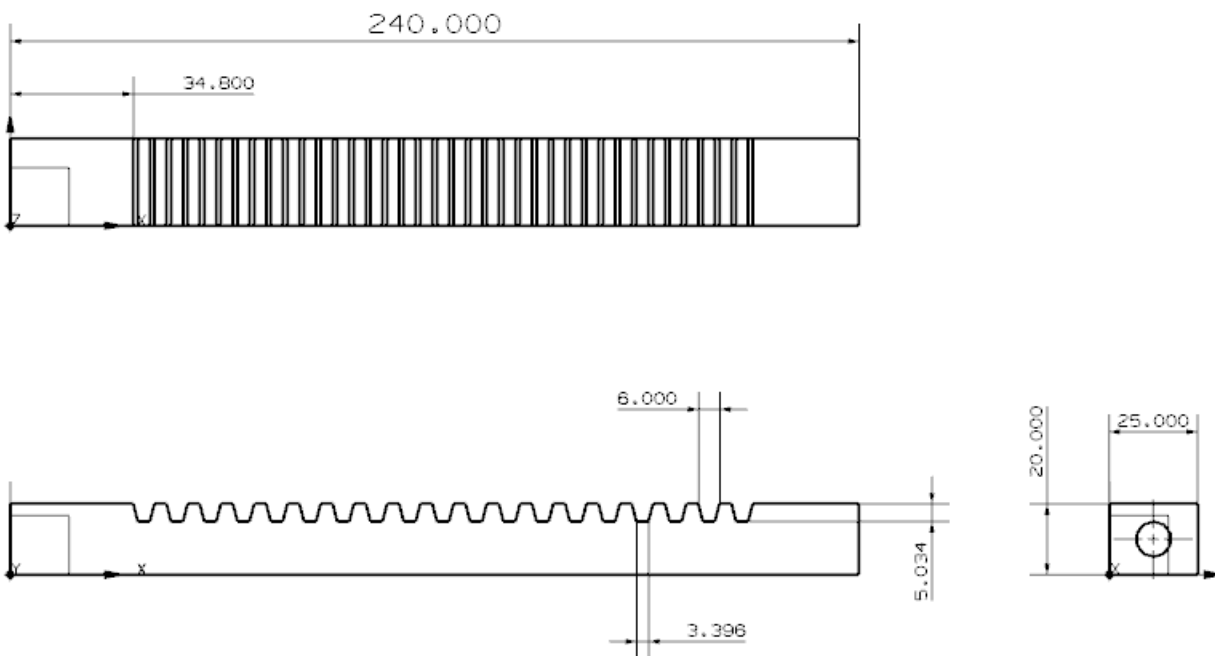


The dimension that represents the distance between these points will appear. You can put the location of the dimension by moving the mouse on the screen. Whenever you place your views in the *Sheet* take into consideration that you will be placing the dimensions around it.

- To set the dimension onto the drawing sheet, place the dimension well above the view as shown and click the left mouse button


Even after creating the dimension, you can edit the properties of the dimensions.

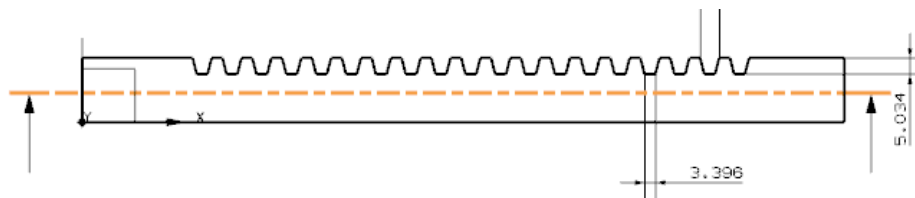
- Right-click on the dimension you just created and choose **Settings** or **Edit Display**
- You can modify font, color, style and other finer details here
- Give dimensions to all other views as shown in the following figure



## 5.4 SECTIONAL VIEW

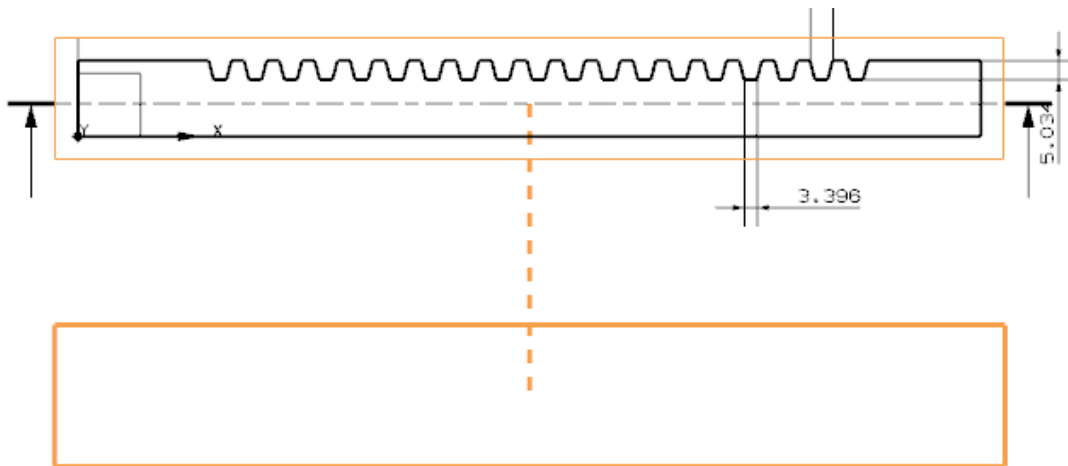
Let us create a *Sectional View* for the same part to show the depth and profile of the hole.

- Choose **Insert** → **View** → **Section** or click the **View Section** icon  from the **View** group in the ribbon bar
- Click on the bottom of the **Base View** as shown in the figure. This will show a **Phantom Line** with two **Arrow** marks for the direction of the Section plane (orange dashed line with arrows pointing upwards).

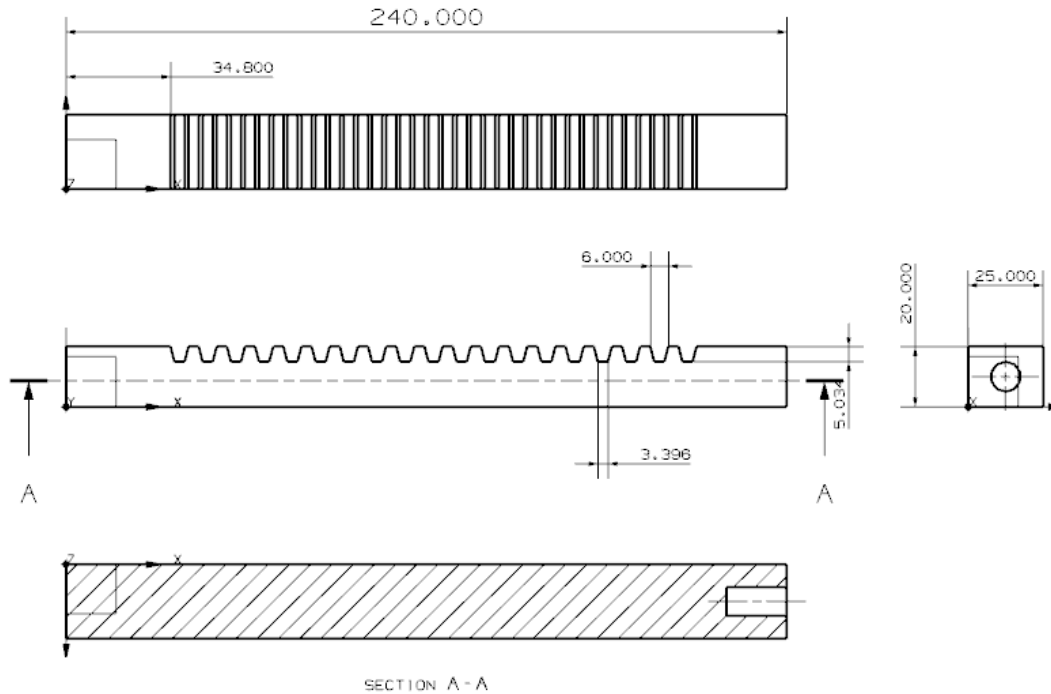


- Click on the middle of the **View** as shown. This will fix the position of the sectional line (Section Plane)

Now move the cursor around the view to get the direction of the **Plane of Section**. Keep the arrow pointing vertically upwards and drag the sectional view to the bottom of the **Base View**.



Adjust the positions of dimensions if they are interfering. The final drawing sheet should look like the one shown in the following figure.



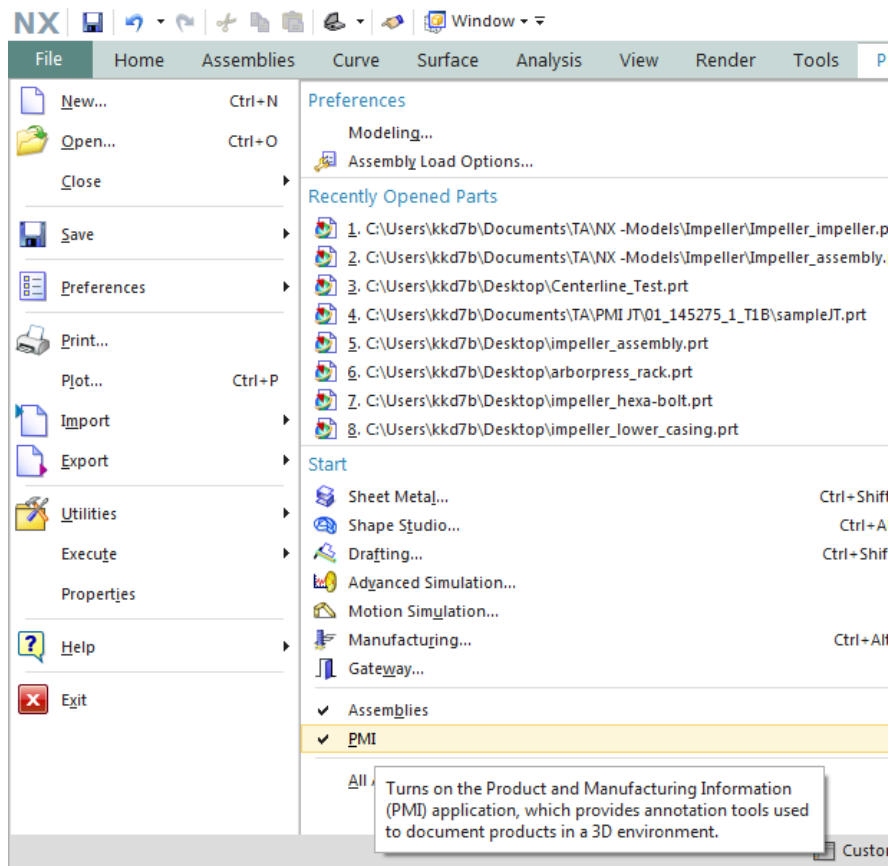
Save and close your model.

## 5.5 PRODUCT AND MANUFACTURING INFORMATION

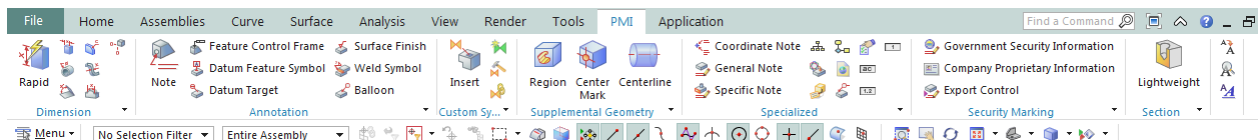
*Product and Manufacturing Information* (PMI) is one of the important applications in NX which provides annotation tools used to document products in a 3D environment. PMI application includes a comprehensive 3D annotation environment that allows design teams to share details such as Geometric Dimensioning and Tolerancing (GD&T), surface finish, welding information, material specifications, comments, government security information or proprietary information, etc. directly to the 3D model. PMI complies with industry standards for 3D product definition and therefore product teams working on collaborative projects would use 3D models as a legitimate method for fully documenting product and manufacturing information.

In the below example, we will open a part file, create dimensions and comments on the 3D model in the PMI application and learn how to inherit the dimensions and comments to the *Drafting* application. This is only for the purpose of illustration.

- Open the file **Impeller\_impeller.prt**
- From the NX 12 interface, choose **File** → **PMI** (turn on the check mark)




This should create an additional tab *PMI* in between *Tools* and *Application* tabs. Select the *PMI* tab to enter the *PMI* application which should look as shown below.



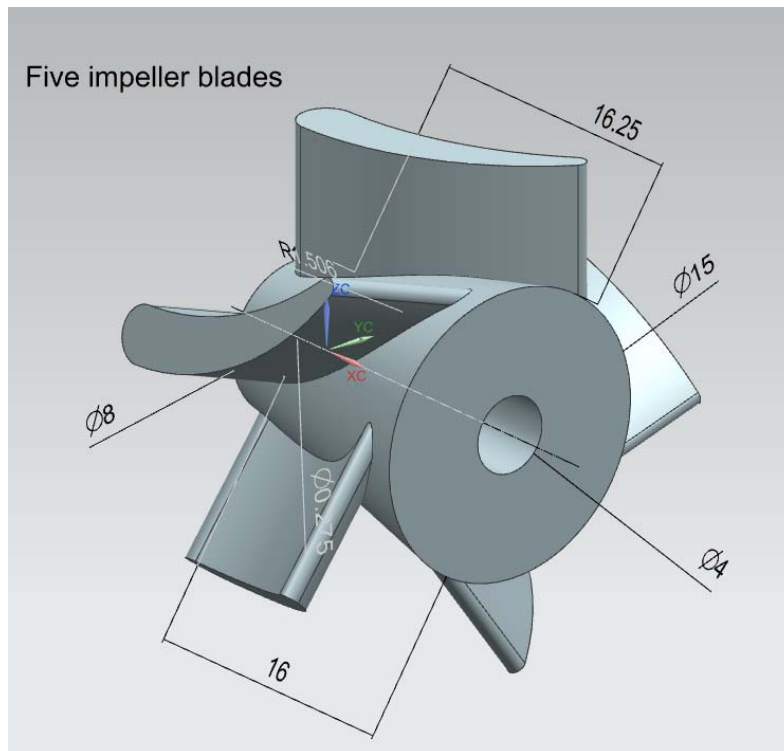
The ribbon bar in this mode would have the *Dimension*, *Annotation*, *Custom Symbols*, *Supplemental Geometry*, *Specialized* and *Security Marking* groups. Each group has several options which could help describe the modeled 3D part. For example, dimensioning options in *Dimension* group, *Surface Finish* and *Notes* in *Annotation* group.



- Click **Rapid** icon
- Select the end surfaces of the impeller as first and second objects to insert the linear dimension or click the **Linear** icon to perform the same task

- Click the **Radial** icon in the **Dimension** group to insert the dimensions of the holes and curved surfaces on the impeller
- Click the **Centerline** icon  in the **Supplemental Geometry** group and select the inner surface of the impeller to insert the centerline for the part
- Click the **Note** icon in **Annotation** group to provide any comments or **Surface Finish** icon, select the object, location of text and leader line to insert the specific surface finish details, if required

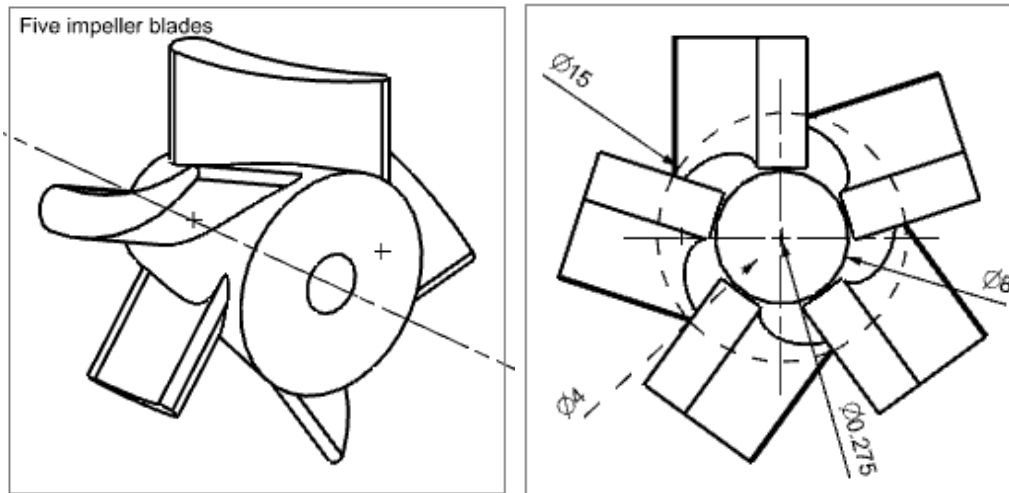
The *Trimetric* view of the impeller after PMI dimensioning would look as shown below.





- Save the file, select **Application** tab and click on **Drafting** icon in the ribbon bar
- Follow the similar procedures explained in the previous section to create the **Drawing** sheet for the 3D part

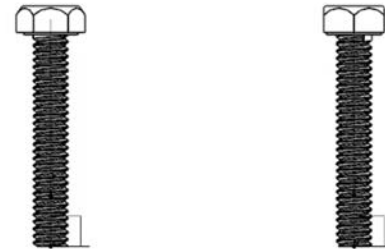
During the creation of the sheet, in the *View Creation Wizard*, select the *Inherit PMI* option, and select the *Aligned to Drawing (Entire Part)* and check the *Inherit PMI onto Drawing* option. This would inherit the dimensions of the 3D model and show on the drawing sheet including the

comments as shown below. The user has to select the appropriate views to reflect the dimensions on the drawing sheet.



## 5.6 EXAMPLE

- Open the model **Impeller\_hexa-bolt.prt**
- Choose **File** → **Drafting** or select **Drafting** in **Application** tab
- On the **Sheet** window, select sheet **E-34 X 44** and change the **Scale** value to **8.0 : 1.0**
- Click **OK**
- Choose **Insert** → **View** → **Base View** or click the **Base View** icon 
- Add the **Front** view by repeating the same procedure explained in the last example 
- Add the **Orthographic Views** including the **Right** view and **Top** view
- Choose **Preferences** → **Drafting**
- Uncheck the box next to **Display Borders** under **View Tab**



The screen will have the following three views.

To visualize the hidden lines,

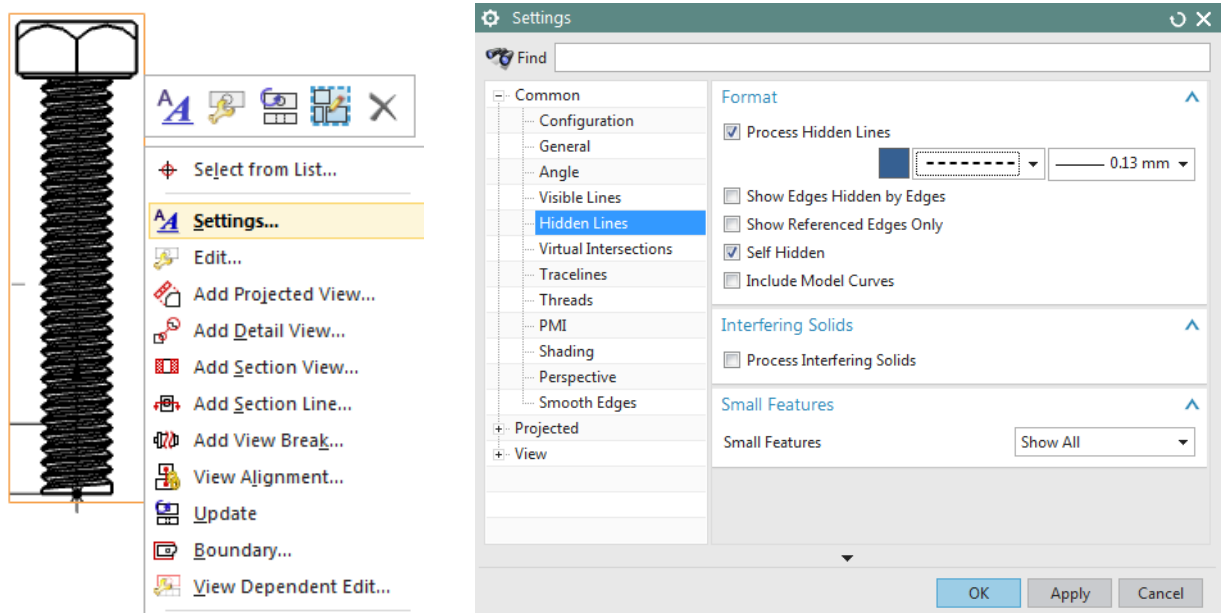
- Choose **Preferences** → **Drafting** → **View**

OR

- Select the views, right-click and choose **Settings** as shown below

A window will pop up with various options pertaining to the views.


- Click on the **Hidden Lines** tab
- Change **Process Hidden Lines** to **Dashed Lines** as shown below and click **OK**

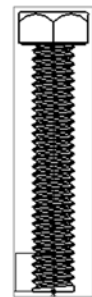
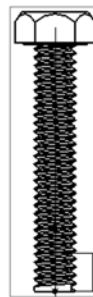


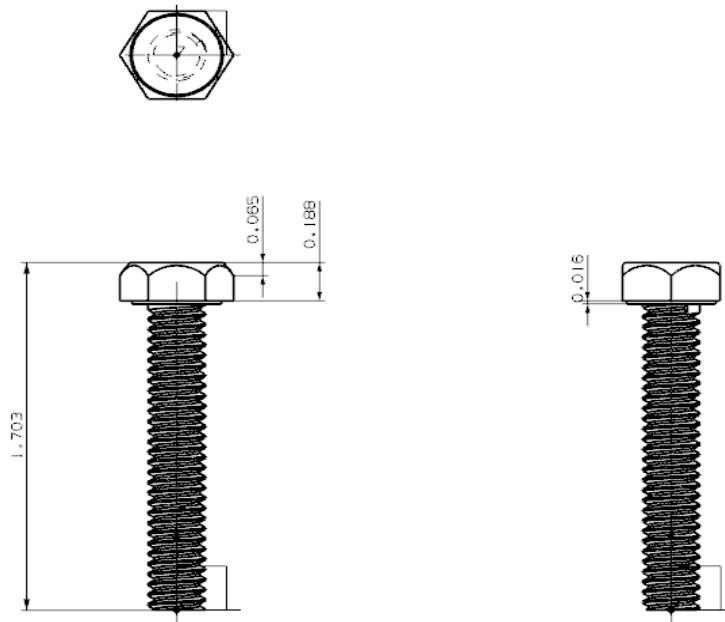
You can see the hidden lines as shown in the picture on the right.



Now we will proceed to dimensioning.

- Choose **Insert** → **Dimensions** → **Linear** or click the **Linear Dimension** icon  in the **Dimension** group
- Give vertical dimensions to all the distances shown below





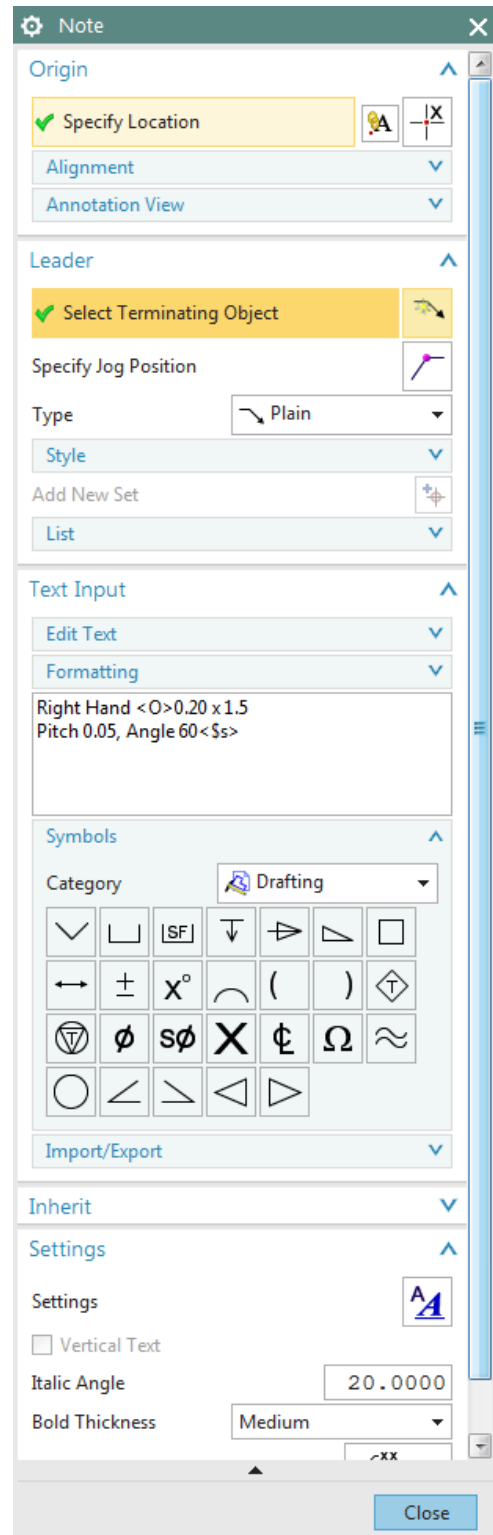
For the threading, we will use a leader line.

- Click on the **Note** icon shown in the Toolbar
- In the **Note** window that opens, enter the following text. You can find Ø and the degree symbol on the **Symbols** tab

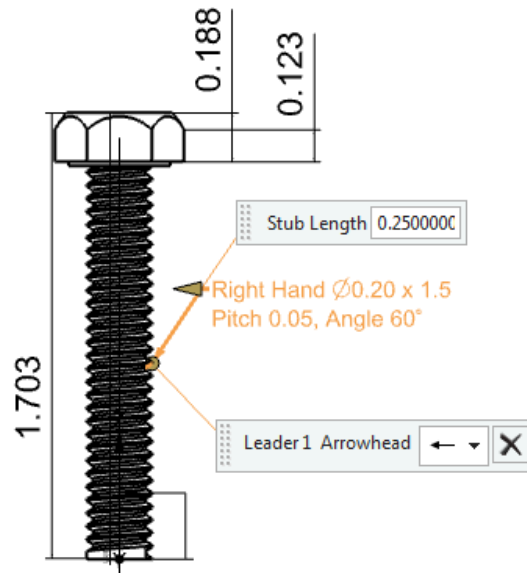
**Right Hand Ø 0.20 x 1.5**

**Pitch 0.05, Angle 60°**

- Click on the threaded shaft in the side view, hold the mouse and drag the **Leader** line next to the view. Let go of the mouse and click again to place the text.









- Close the **Annotation Editor**

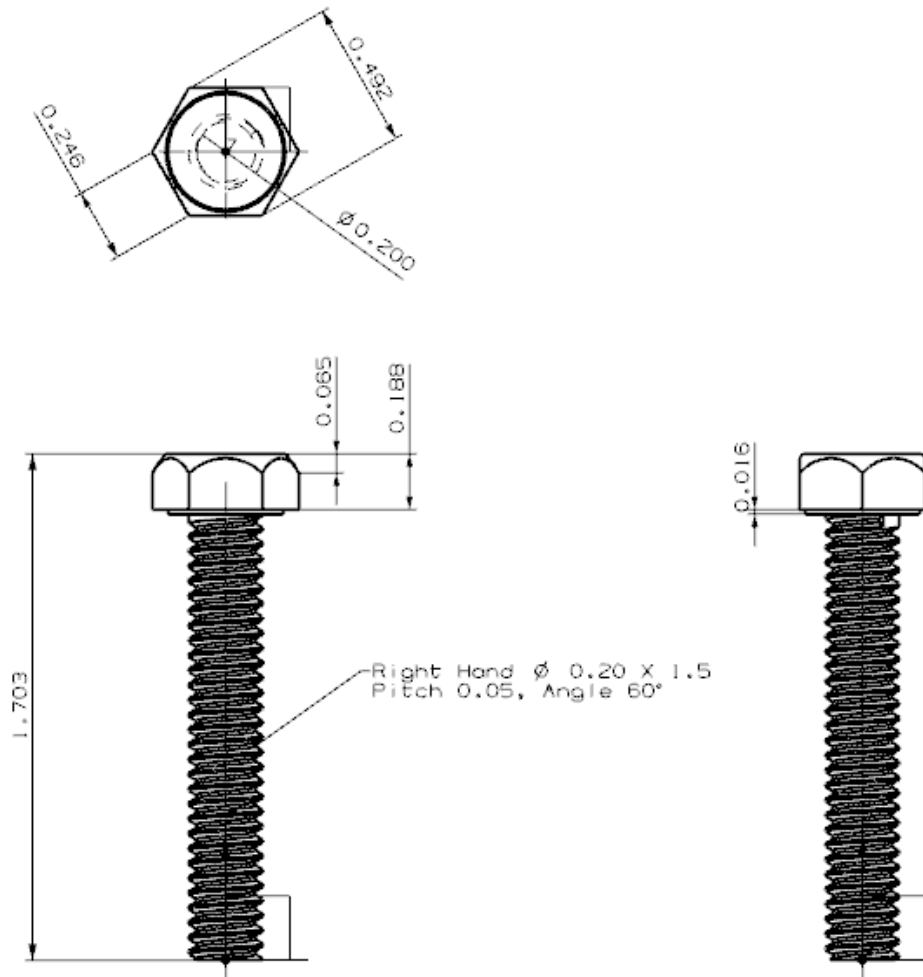
Since the height of the *Lettering* is small, we will enlarge the character size as well as the arrow size.

- Right-click on the **Leader** and select **Settings**
- Click on the **Lettering** tab
- In the **Text Parameter** section, increase **Height** to make the leader legible
- Click on the **Line/Arrow** tab
- In the **Format** section, increase the **Length** of the **Leader**

Now we will add additional dimensions and views.

- Choose **Insert** → **Dimensions** → **Radial** or click the **Radial Dimension** icon  in the **Dimension** group
- Click the circle of the bolt in the top view to give the diameter dimension
- Click **Insert** → **View** → **Base View** or click the **Base View** icon 
- Select the **Isometric** view and place the view somewhere on the screen

The final drawing is shown below. Remember to save.



## 5.7 EXERCISE

Perform *Drafting* and annotate dimensions for the parts that you modeled in chapter 4.